

## APPLICATION OF CFD CODES TO THE DESIGN AND DEVELOPMENT OF PROPULSION SYSTEMS

W. K. Lord, Project Engineer  
 G. F. Pickett, Senior Project Engineer  
 G. J. Sturgess, Senior Research Engineer  
 H. D. Weingold, Senior Research Engineer

UNITED TECHNOLOGIES CORPORATION  
 Pratt & Whitney  
 Engineering Division  
 400 Main Street, East Hartford, CT 06108

## SUMMARY

The internal flows of aerospace propulsion engines have certain common features that are amenable to analysis through computational fluid dynamics (CFD) computer codes. Although the application of CFD to engineering problems in engines has been delayed by the complexities associated with internal flows, many codes with different capabilities are now being used as routine design tools. This is illustrated by examples taken from the aircraft gas turbine engine of flows calculated with potential flow, Euler flow, parabolized Navier-Stokes, and Navier-Stokes codes. Likely future directions of CFD applied to engine flows are described, and current barriers to continued progress are highlighted. The potential importance of the Numerical Aerodynamic Simulator (NAS) to resolution of these difficulties is suggested.

## INTRODUCTION

If an aerospace propulsion system is taken as any continuous-flow engine powering a vehicle for flight through the atmosphere, then included by this definition are gas turbines, ramjets, scramjets and liquid-fueled rocket motors. Although these engines look and operate very differently, they have certain common features that are amenable to analysis, e.g., flow turning, fuel injection, mixing, combustion, radiation, etc. In addition, certain of them have common flow systems (such as labyrinth seals, rotating cavities, cascade flow, stator-rotor blade interactions, diffusers, nozzles, etc.) that also can be addressed. Propulsion systems, with the exception of rocket motors, are usually enclosed in some kind of aerodynamic fairing. Flows over, and into and out of, these fairings form a critical part of the whole, and provide a link between the external aerodynamics of the vehicle and the internal fluid dynamics of its powerplant. It is flows such as these with which internal computational fluid dynamics (CFD) is concerned.

Computational fluid dynamics is emerging as a vitally important tool in the design and development of aerospace propulsion systems. It is a tool that is being used extensively at present, and its continued growth for these applications is assured since it makes available to the designer and development engineer information that can be provided in no other way. This use and potential of CFD is demonstrated in the next section through examples taken for convenience, from the gas turbine engine. It will be shown that efforts to obtain realistic and quantitatively accurate simulations are confronted primarily by two technical barriers. They are: (a) the availability and affordability of the computer capacity, and (b) the ac-

curacy of the physical models that need to be incorporated into the CFD codes. It is anticipated that the existence and operation of the Numerical Aerodynamic Simulator (NAS) will help to push back the first of these barriers and to permit the extensive code validation against benchmark experimental data that eventually will generate improved physical models and so, will push back the second barrier.

The application of CFD techniques in the aerospace industry has been very successful in external aerodynamics. However, although the status of CFD code development is equivalent, successful application to complex internal flows as an engineering tool has not yet advanced to the same degree, and the engine manufacturers appear to lag the airframe manufacturers in exploiting CFD by a period of at least ten years. It is only in the last five years or so when any significant use of CFD in powerplant design and development has taken place. However, recent progress in the area has been rapid.

Much of the reason for the delay in the use of CFD codes as an engineering tool in the design and development of propulsion systems can be attributed to the considerably more difficult task associated with calculating internal fluid flows. Many of the physical simplifications and computational shortcuts applicable to problems in external aerodynamics are just not suitable for analyzing internal aerodynamic problems. Two-dimensional potential, and two and three-dimensional Euler equation CFD codes, which form the cornerstone approach for analyzing external aerodynamic flows, are used extensively to analyze turbomachinery flows; but in this internal application, they need to be calibrated to account for viscosity-generated blockage and airfoil trailing edge conditions. Boundary layer codes and inviscid/viscous interaction codes are used in conjunction with the potential and Euler equation solvers to determine estimates of airfoil heat transfer and to define regions of possible separation. However, at off-design conditions where the inviscid/viscous interactions cause separated regions, this approach is not adequate. Parabolized Navier-Stokes codes can be used for a class of problems provided that regions of separated flow are very small, and there exists no significant recirculation of flow. There are many situations, however, where the internal aerodynamics is dominated by secondary flow development, and such large regions of separated flow can only be addressed through solution of the full Navier-Stokes equations with the attendant costs and computer demands. In almost all cases, the enclosing geometries are extremely complex.

This is very much the case for flows in combustors and dump diffusers, regions of flow on the

endwalls near leading edges of compressor and turbine airfoils, and engine secondary flow systems such as rotating cavities and labyrinth seals. These problems can only be addressed through the solution of the full Navier-Stokes equations. Because the bounding geometries in internal aerodynamic flows are invariably complex and the flow fields are also, large numbers of grid points are needed for adequate resolution of gradients; this gives rise to high computing demands and the attendant costs. The perceived difficulties of calculating internal flows in a realistic and affordable manner did inhibit serious acceptance of CFD until such time as computer developments made the difficult task feasible. In practice, pragmatic engineers in the propulsion field utilize whatever solution-approach is physically adequate for their purposes and computationally efficient.

The application of CFD codes in the propulsion industry will be demonstrated for a variety of design problems associated with the aircraft gas turbine engine. It will be shown that some cases can be adequately analyzed with the relatively simple CFD codes, while other cases require the application of a full Navier-Stokes code. Similar examples could also be presented for liquid rocket motors and ramjet engines.

The authors wish to thank the Pratt & Whitney Engineering Division of United Technologies Corporation for permission to present this paper. The views expressed are those of the authors.

#### CURRENT APPLICATION OF CFD CODES

In order to illustrate the current state of the art, various applications are presented under the headings of specific CFD code types.

##### Potential Flow Codes

Since the mid-1970s, subsonic and transonic potential-flow codes have been widely used at Pratt & Whitney to analyze two-dimensional compressor or turbine airfoil cascades, and axisymmetric inlet flow fields. In the last five years, there has been an increasing shift toward three-dimensional flow analyses.

One area in which three-dimensional potential methods are extensively used is nacelle aerodynamics. Nacelle-related problems, such as the design of inlets and nozzles, involve a combination of external and internal aerodynamics. Since the external flow about the nacelle is essentially irrotational, the potential-flow approximation is valid for many of these problems. It is particularly applicable to the aerodynamic analysis of inlets and fan cowls on turbofan engine installations for subsonic transport aircraft. (It is not a very good approximation of the physics of nozzle flow fields, however, because of the rotational engine exhaust flow, strong shocks, and turbulent shear layers typical of nozzles at high pressure ratio.)

Panel methods have seen extensive use in the airframe industry for a number of years. They have been used at Pratt & Whitney to obtain potential solutions for the flow about complex geometries such as the installed turbofan nacelle (Lord and Zysman, 1986). Most recently, they have been applied to the aerodynamic design and analysis of

Prop-Fan nacelles. The Pratt & Whitney/Allison Gas Turbine/Hamilton Standard Prop-Fan propulsion system is a geared counter-rotating pusher. The nacelle (figure 1) has an auxiliary air intake for the gearbox air/oil heat exchanger. The engine exhaust gas exits the nacelle through discrete lobed nozzles located upstream of the propellers. Panel codes have proved to be quite useful in analyzing the flow into the auxiliary air intake and around the lobed nozzles (figure 2). Pratt & Whitney is currently using primarily the VSAERO code (Clark, et al, 1984). VSAERO is a second-generation panel code that exhibits very little leakage, and is thus well suited to problems involving combined external/internal aerodynamics.

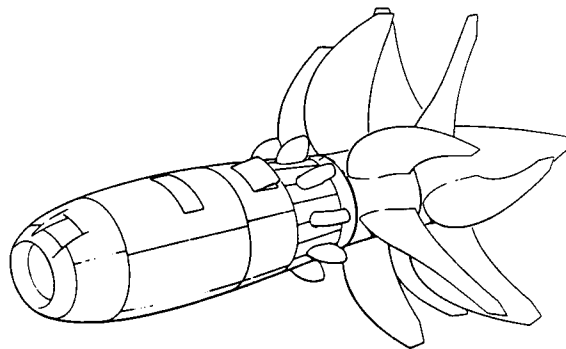


Figure 1 Arrangement of P&W/AGT/HS Pusher Prop-Fan

##### LOBED NOZZLE WITH FAIRING MACH 0.76

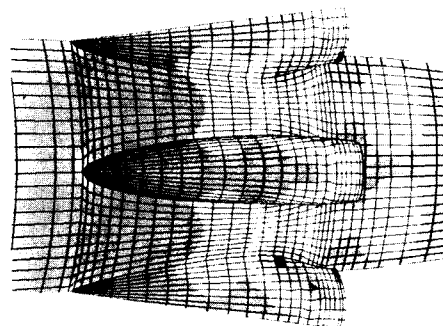


Figure 2 Distributions of Mach Numbers Over Lobed Nozzle of Pusher Prop-Fan - Panel Method (Original in Color)

The significant advantage of panel codes, of course, is that they do not require an off-body computational grid. They can handle completely arbitrary geometries, as opposed to finite-difference or finite-volume solvers for which the grids and boundary conditions are typically set up for a specific component. The computer storage and processor time requirements for panel codes are relatively modest, well within the capacity of current machines. One major drawback is that current panel-method solutions are strictly valid only in the subsonic, and not in the transonic, flow regime.

## Euler Equation Flow Codes

Three-dimensional potential methods are not generally applicable to propulsion system internal flows because of rotational effects (e.g., spanwise total temperature and total pressure profiles). This fact helps to explain the intensive development effort at Pratt & Whitney of three-dimensional (3-D) Euler methods for fan/compressor and turbine aerodynamics.

Euler equation codes are being applied to the design and analysis of all the major components of the gas turbine engine with the exception of the combustor. The particular version used at Pratt & Whitney is that developed by Ni (1982). With good input/output subroutines, and the incorporation of multi-grid methods to speed-up the calculation, the Ni Euler-equation solver is extensively used by the engineer as just another design tool. The Euler code is principally used to determine the surface loading distributions on airfoils, endwalls and nacelles. It is also used, however, in conjunction with 2-D and 3-D boundary layer analyses to identify potential separated flow regions and high heat load regions in turbine airfoil passages.

The code was initially developed in two-dimensional form to calculate the blade-to-blade pressure distributions in turbine and compressor airfoils. Its main advantage over potential flow codes is that it can be applied well into the transonic flow regime. The code was extended to three-dimensional steady flow for application to turbine airfoil passage flows. In addition, because of the relatively small viscous/inviscid interactions that occur in a well-designed turbine airfoil passage away from endwalls, the code has proved to be an excellent tool for predicting airfoil loadings as a function of span for a variety of three-dimensional turbine designs.

The Ni code demonstrated early that problems in existing highly loaded airfoil designs could be avoided by changing the airfoil section stacking and also the sections to obtain improved pressure distributions. In this way, local diffusing regions could be avoided, and as a consequence, improved performance invariably was achieved. Some examples of this are presented in an American Institute of Aeronautics and Astronautics paper by Huber, et al (1985).

One of the biggest problems in applying a three-dimensional Euler solver to the design of turbomachinery airfoils is the definition of the upstream and downstream boundary conditions for each airfoil row. An approach adopted to overcome this problem was to develop a multi-stage calculation procedure, where the upstream boundary condition is defined ahead of the first vane and the downstream boundary condition is defined behind the last blade row. Interface planes between adjacent blade rows are defined, and flow conditions at these planes are calculated as part of the solution. Implementation of this approach, however, requires that certain simplifying assumptions be made. Since the flow in actual turbines is highly unsteady, and because invariably, for dynamic structural reasons, the number of airfoils in neighboring rows is different, the flow field at the interface planes has to be circumferentially

averaged before transmitting the information to the calculation of the neighboring airfoil row which, of course, is in a different frame of reference. In principle, the calculation procedure could be set up to include an integral number of neighboring blades and vanes; and the Euler code, being a time-marching scheme, could handle the unsteady boundary conditions at the interface plane due to the need to change from stationary to moving frames of reference between vanes and blades and vice versa. However, the current state of the art of computers does not allow mesh densities that would be sufficiently large to accurately capture the flow. It should be emphasized, however, that the relatively high levels of turbine performance have been obtained with time-averaged data and based on the demonstrated accuracy of predicting "mean" pressure distributions and flow angles. As such, the multi-stage analysis with circumferentially averaged interface plane conditions has provided a significant step in simulating the mean flow effects for three-dimensional, rotational flow fields in turbines.

Figure 3 shows a view of a typical two-stage high pressure turbine computational mesh. There are a total of four airfoil rows contained within the computational domain (i.e., two vanes and two blades). The lines formed by adjacent airfoil row mesh planes are the "interface" boundaries that subdivide the computational domain into four sectors. The three-dimensional flow solution for the resultant multisector problem is determined by applying the code to all the sectors and imposing an interior flow boundary condition to the interface mesh points at each time iteration step. The far-field boundary conditions for this multistage computational procedure then become the first vane upstream conditions and the second blade downstream conditions.

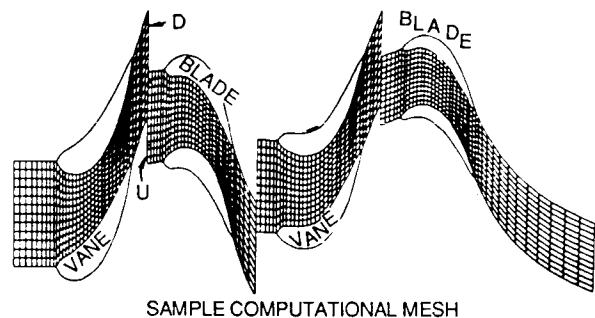


Figure 3 Computational Mesh for Turbine Vane/ Blade Rows

Several calculations of two-stage turbines have demonstrated the accuracy of predicting airfoil and endwall pressure distributions and exit flow angles from each airfoil row. In order to achieve this accuracy, the secondary flow within the passages needs to be reasonably well represented. Figure 4 illustrates the predicted vortex structure that can exist in the first blade depending on the flow field exiting the first vane. Such secondary flow can have a large effect on the flow angles exiting the blade row.

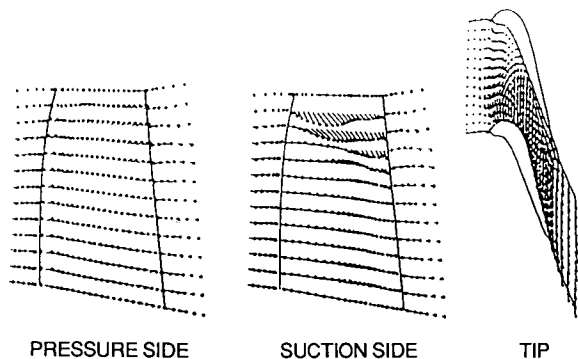


Figure 4 Predicted Secondary Flows in First Blade Passage - Euler Code

As the capacity of computers increases, multi-stage Euler codes will be extended to simulate inviscid unsteady vane/blade interaction flows, and viscous terms will be added to the governing equations so that first order viscous effects will be simulated in the calculations.

The Ni Euler code has also been used to compute the inviscid flow about inlets at angle of attack. In this case, a uniform static pressure boundary condition is applied at an internal station at or just downstream of the actual fan-face station. At low inlet air flows, the flow is isentropic and the relationship between the captured mass flow and the fan-face static pressure may be calculated directly from the one-dimensional isentropic relations. At flow approaching the maximum inlet air flow capacity, however, strong internal shocks can occur, particularly at high angle of attack (figure 5). In the case of nonisentropic flow, a static pressure boundary condition is specified; but the corresponding actual inlet mass flow must be obtained from the computed solution by integration of mass flux across an internal control surface. A mass-averaged inlet total pressure loss (shock loss) is also obtained by integration of entropy flux across the control surface. It should be noted that accurate prediction of shock loss requires an Euler algorithm in which artificial dissipation effects, which lead to spurious numerical total pressure loss, are minimized.

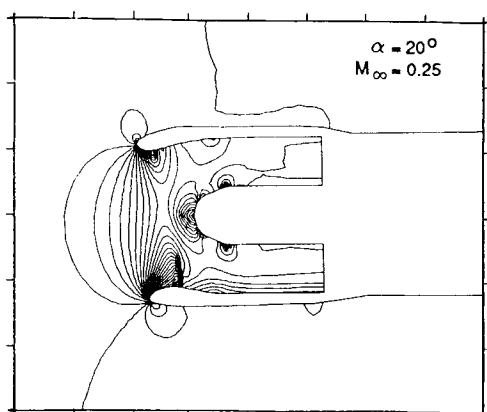


Figure 5 Mach Number Contours for Engine Inlet Showing Shock Formation - Euler Code

Parameters of interest from a design standpoint at high inlet air flow are inlet total pressure recovery and maximum inlet flow capacity. Inlet recoveries can be crudely estimated by simply adding the shock loss from the Euler calculation to the boundary-layer loss obtained from a separate strip boundary-layer calculation. A comparison with model test data (figure 6) indicates that recovery levels are fairly well predicted and that the maximum air flow capacity, due to choking at the inlet throat, is predicted quite accurately.

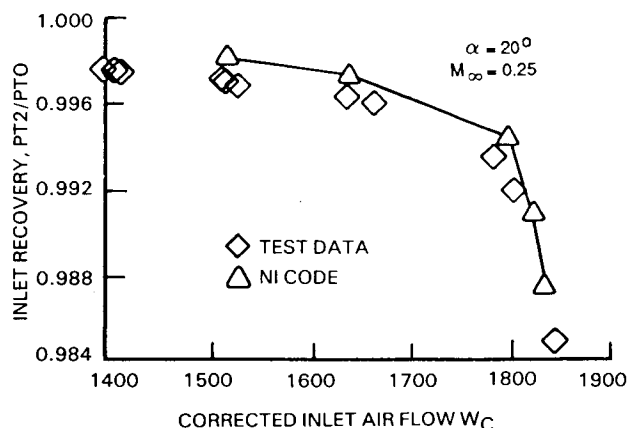


Figure 6 Comparison of Predicted and Measured Inlet Pressure Recoveries - Euler Code

#### Parabolized Navier-Stokes (PNS) Codes

The Parabolized Navier-Stokes (PNS) approach has been applied to both subsonic and supersonic/hypersonic flow fields. At Pratt & Whitney, the emphasis has been on development of a PNS code for subsonic internal flows in three-dimensional ducts of arbitrary cross section and curved centerline. A development version of this code is currently undergoing extensive calibration. The expected applications include fan ducts in turbofan nacelles, the radially-offset exhaust duct for the lobed nozzle in the Prop-Fan nacelle, the circular-to-rectangular transition duct in military two-dimensional nozzle exhaust systems, and several components in the Space Shuttle Main Engine.

In the PNS formulation, upwind differencing is used for transport variables, streamwise diffusion terms are neglected, and the streamwise pressure gradient is prespecified. In the case of the Pratt & Whitney PNS code, global elliptic effects are incorporated through use of a coarse-grid Euler solution; i.e., it is this solution that provides the background pressure gradients. The reduced set of PNS equations allows streamwise marching of planes of data in space, as opposed to time marching of three-dimensional data blocks for a full Navier-Stokes solver. This results in an order-of-magnitude reduction in both computer core memory and central processing unit (CPU) time requirements.

The PNS code has been applied to a well-known example case, the Boeing 727 center-engine S-duct inlet (Kunz and Rhie 1986). The computational grid and a comparison of the Euler and PNS pressure distribution with experimental results from model

test (Ting, et al, 1975) are shown on figure 7. The pressure distributions are here expressed in terms of an equivalent Mach number based on upstream total pressure. The inviscid Euler solution agrees well with the data over the forward portion of the duct but underpredicts the Mach number levels in the aft portion of the duct. This is as expected; the boundary layer builds up with distance down the duct and the displacement effect of the boundary layer raises the local Mach number levels. The PNS code viscous solution reflects the boundary-layer displacement effect and gives better agreement with the data in the aft portion of the duct.

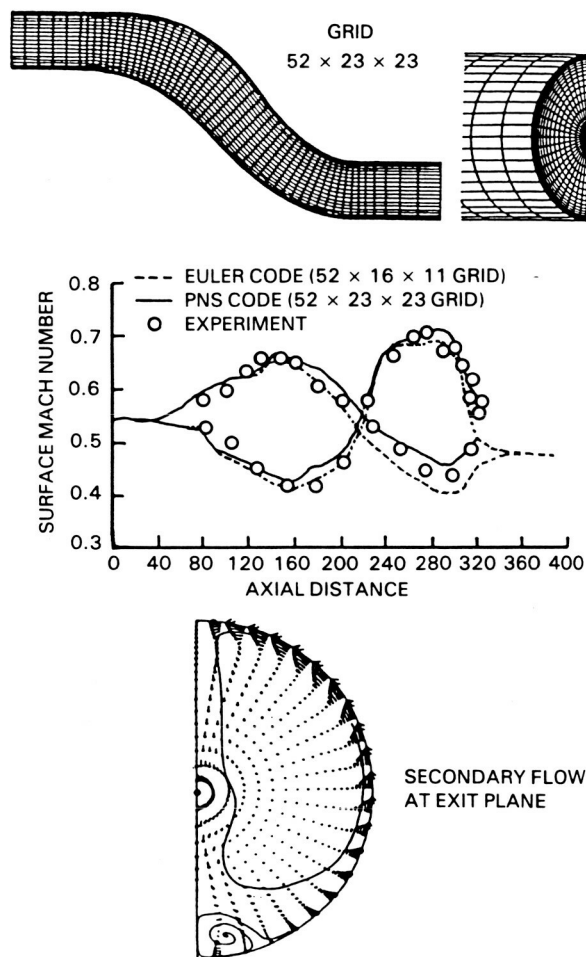


Figure 7 Calculation of B727 Center-Engine S-Duct Inlet Showing Secondary Flow Development - PNS Code

The predicted secondary flow pattern at the exit plane is also illustrated in figure 7. The cross-channel pressure gradient induced by the second bend drives a circumferential flow in the boundary layer toward the top of the duct. A much smaller region of secondary circulation is evident at the bottom of the exit plane cross section; this is the remnant of the secondary flow induced by the first bend in the duct. It shows up on a total pressure contour as a region of high loss.

#### Navier-Stokes Codes

Flows with dominating recirculation demand

solution of the full Navier-Stokes equations. Examples of such flows are rotating cavities, labyrinth seals and combustion chambers. Two examples of such flows are presented.

Most practical internal fluid flows of interest are turbulent. This is usually dealt with through a simple turbulence management strategy in which the Navier-Stokes equations are written in Reynolds-averaged or Favre-averaged (density-weighted) form and are solved as a stationary flow. The turbulent fluxes introduced into the equations by the averaging are represented through the eddy viscosity hypothesis and gradient diffusion. Turbulence modeling is used to find relationships for the eddy viscosity. This modeling may be appropriate for both high and low turbulence Reynolds numbers, or a high Reynolds number model may be used together with turbulent wall functions for the near-wall region. For variable density and chemically reacting flows, the energy equation and species transport equations are included. The partial differential equations are arranged into a general form that consists of convection and diffusion terms, and source terms describing the generation and dissipation of the dependent variable. A heat release rate expression is provided for reacting flows. Finite differencing is used to discretize the equations. Since pressure does not appear directly in the momentum equations or the continuity equations but is a dependent variable nonetheless, an algorithm is used to obtain a pressure field and correct the velocity field in such a manner that the continuity and momentum equations are simultaneously satisfied. An implicit solution algorithm is used, with the initial guesses for the field variables being iteratively updated until convergence is reached (Sturgess, 1983). The liquid fuel is accounted for through a Lagrangian spray model that is coupled with the Eulerian gas flow field through a particle-source-in-cell technique (Sturgess, et al, 1985b).

Part of the difficulty of presenting internal flows is that they are totally enclosed. In addition, the enclosing geometry can be extremely complex. The combustion chamber of the gas turbine engine is one such component; this is illustrated on figure 8 which is a view of an annular combustor presented from an interior perspective. Although the flow can be considered as periodic about fuel injectors, it may be appreciated that definition of the many streams (film cooling air, combustion air jets, dilution air jets, dome cooling air, swirler air, and liquid fuel), entering the computational domain, as well as the basic bounding geometrical contours, demand a large computer resource. Even when reduced to a repeating segment, the combustor geometry presently has to be greatly simplified, an example of which is given in figure 9.

Calculation of the simplified combustor of figure 9 yields a tremendous amount of useful information concerning the details of the reacting flow contained therein. Figure 10 gives an isotherm contour plot at a cross-section in the dome and shows details of the film cooling, the dome cooling associated with the heatshield, and the results of local air admission around the fuel injector. Note that the temperature field is not symmetric about the axisymmetric fuel injector and heatshield geometries; this is due to the swirling

air introduced by the airblast fuel injector. An additional cross-section is contained in figure 11. This section is taken at a plane through the second row of air-addition ports. Streaklines are provided as motion-cues and are colored with the appropriate color for the local temperature field. The complex character of the flow field can be appreciated.

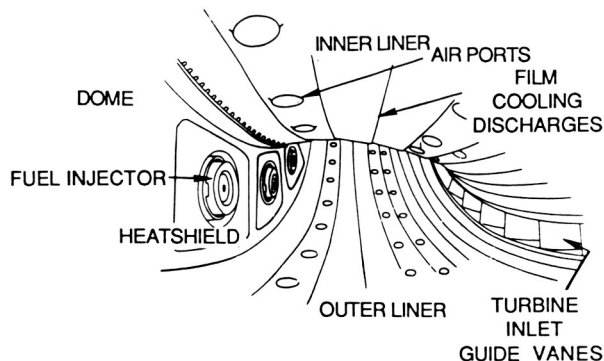


Figure 8 Interior View of an Annular Combustor for a Gas Turbine Engine

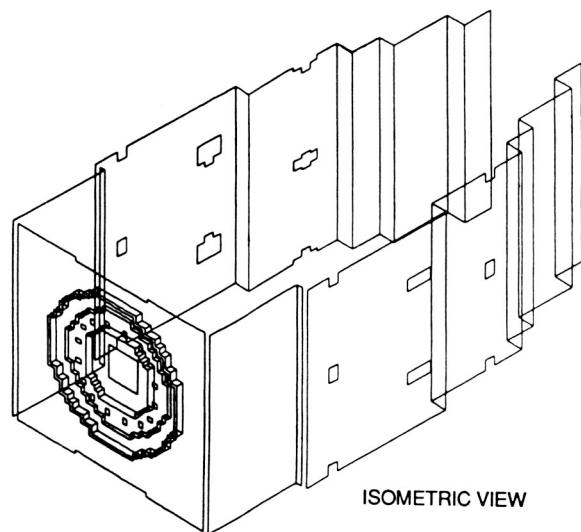


Figure 9 Simplified Repeating Sector of an Annular Combustor

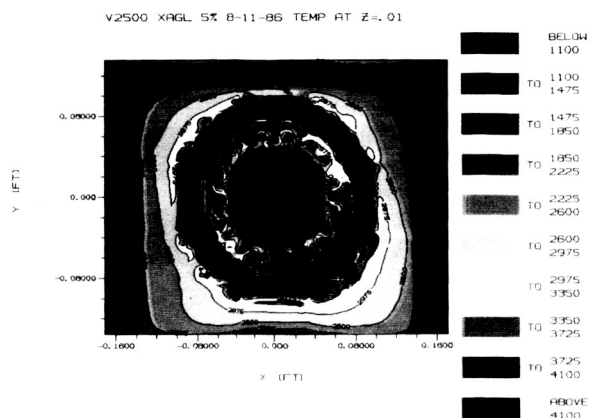


Figure 10 Isotherm Contour Plot Close to Dome in Simplified Dome - NS Code (Original in Color)

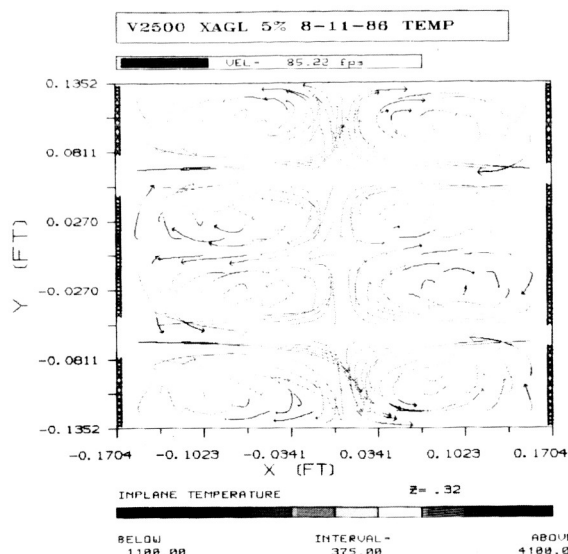


Figure 11 Combined Streakline/Temperature Plot at Cross-Section Through Second Row of Air-Ports (Original in Color)

Figure 11 demonstrates the dominant characteristic of combustor flows, i.e., they are wholly elliptic. This is best illustrated by means of animated streaklines, of which figure 12 is a "still" rendering.

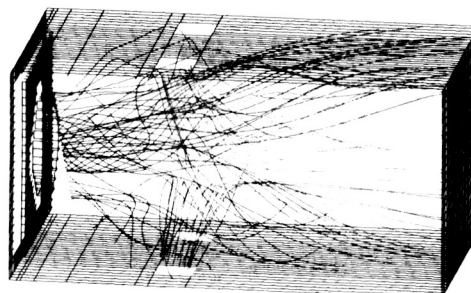


Figure 12 Isometric Projection of "Frozen" Animated Streaklines in a Model Primary Zone (Original in Color)

Calculations in the combustor such as those shown are most useful to understand the flow processes, for determining the origin of hot-spots on the confining liners and dome (Sturgess, 1980), and in investigating geometric changes to develop the outlet temperature distributions. This is all done by directly viewing the reacting, hot-gas flow field, rather than by measuring its effects on the confining boundaries and then postulating a field which is to be modified in some way.

The calculations shown were performed with a Cartesian coordinate system on a 50 x 40 x 41 grid, requiring a storage of 30 megabytes and about 11 CPU hours on an IBM 3090 (fast scalar) computer to reach a somewhat arbitrary convergence level of 5 percent (residual source sum). At this level, the definition of the combustor is inadequate, and calculations at the 200 megabyte level are being explored, although it is estimated that 300 megabytes might be necessary for adequate representation of the geometry and resolution of the flow gradients. The solutions currently obtained cannot be considered to be grid-independent. Thus, when calculations like this are used

for parametric diagnostic studies, all variations contemplated must be calculated on the same grid; this necessitates very careful thought and advanced planning in establishing the initial grid.

The accuracy of combustor simulations is being determined at present not by the numerics or the physical modeling but by computer capacity and the cost of solution.

Prepared originally for the combustion chamber, the availability of a general Navier-Stokes code written in modular form containing a suite of switchable physical models and an arbitrary geometry capability, enables application to be made to internal flows with recirculation that arise in many areas of the gas turbine engine, including secondary systems as well as the main gas path. An example of such an important secondary system is the internal cooling of turbine aerofoils where there is a need to understand and predict the effects of rotation on local heat transfer and pressure loss, especially for blades that have multipass coolant passages. Accurate calculation of these quantities is very important as it can lead to an improvement in the reliability of predicting turbine airfoil temperatures and, ultimately, blade life. Accurate estimates of temperature and life permit available cooling air to be used most effectively and reduces its negative impact on turbine performance.

In order to demonstrate the potential for this, a simplified geometry of a blade passage was calculated, as shown on figure 13 (Sturgess and Datta, 1987). It consisted of a single outflow leg and an inflow leg. The grid was a coarse  $33 \times 11 \times 5$ , and these demonstration calculations were not considered to be grid independent, and the resolution is low. For the baseline calculation of 600 RPM, the Reynolds number was 30,000 and the Rossby number was 0.174; the passage aspect ratio was unity and its length-to-width ratio was 12.

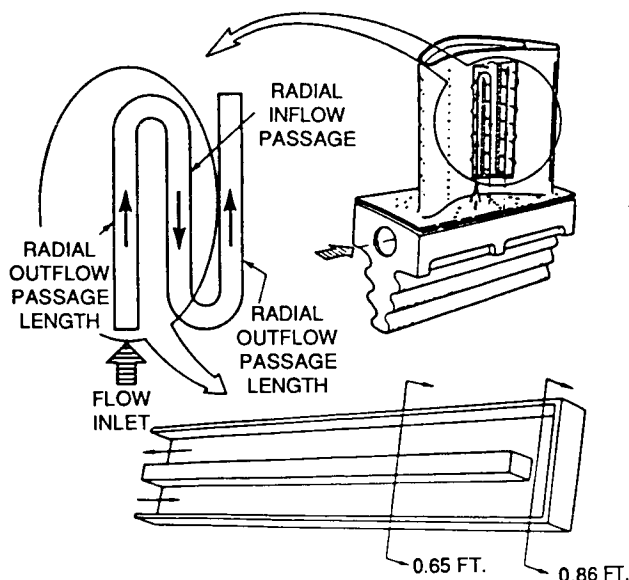


Figure 13 Schematic of Simulated Turbine Blade Coolant Passage

Figure 14 gives the flow field development along the passages at 600 RPM and at the two

cross-sections shown in figure 13 for several rotational speeds. The flow visualization technique is by streaklines, and results for four rotational speeds of 0, 60, 600 and 1900 RPM are presented. The development of secondary flows due to the influence of Coriolis forces can be seen. In the outflow (away from the axis of rotation) leg of the passage, a pair of counter-rotating vortices develop. The vortex centers are shifted slightly from the centerlines of the leg towards the pressure side, but the size and strengths of the vortices are about equal. As the vortex pair enters the turn, their rotational velocity is overcome, and the flow changes direction as mass is forced to the pressure side of the turn. The flow entering the passage inflow leg from the turn is therefore forced into a right-angled corner. As it escapes from the corner to begin flowing down the inflow leg (towards the axis of rotation), the air has no choice other than to establish a single vortex. The action of the turn is thus to coalesce the vortex pair formed by Coriolis forces into a single vortex completely filling the passage inflow leg. Other sections of the passage show that the double vortex structure of the original system begins to reestablish itself as the flow proceeds down the inflow leg towards the axis of rotation, although initially this vortex pair is not symmetrical, of course. The flow behavior observed is intuitively correct.

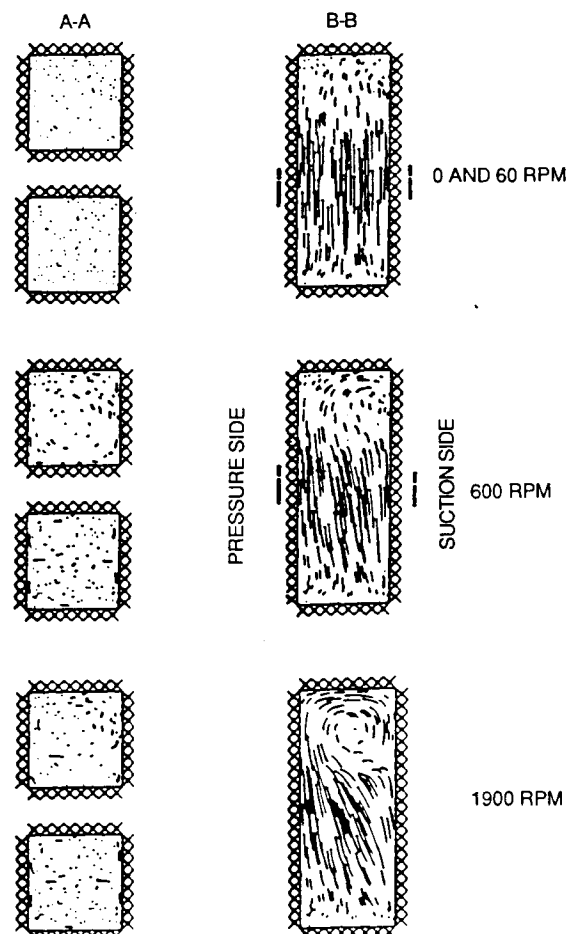


Figure 14 Flow Visualization in Simulated Coolant Passage for a Range of Rotational Speeds - NS Code



Although the code can give adequate quantitative accuracy in three-dimensional rotating passages of very simple geometry, as was established by subsequent calculations (Sturgess and Datta, 1987), the geometries of practical interest require more computer capacity than is currently available. For example, the use of trip-strips in the passages to enhance the heat transfer requires that the induced disruption of the laminar sub-layer in between strips be both calculated and resolved in detail. Such a calculation demands the use of appropriate low Reynolds number turbulence models and many grid lines adjacent to the walls. Again, progress is limited by the available computer facility.

#### FUTURE DIRECTION

It has been shown by way of example, how current CFD codes are being used in the design and development of aerospace propulsion systems. In conjunction with highly refined design criteria, advanced materials and advanced manufacturing technology, gas turbine component performances have progressively increased so that the thrust specific fuel consumption (TSFC) of commercial and transport gas turbines has been reduced at a rate better than 1% per year over the past two decades (figure 15). Also shown in figure 15 is that, with the introduction of advanced propulsion concepts like the Prop-Fan or the ducted fan, the potential for continued significant TSFC reductions will continue. Improved modeling of the gas path flows using the developing CFD codes in the new propulsion systems and in advanced versions of existing gas turbine engines will play an important role in achieving the full potential of these propulsion systems.

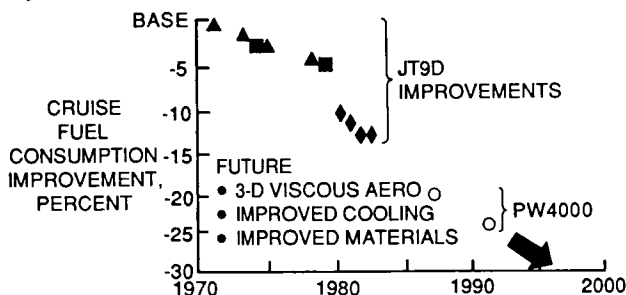


Figure 15 Improvement in Gas Turbine Engine Fuel Consumption with Time

For military gas turbine engines, high thrust-to-weight ratio and specific thrust are major requirements. Figure 16 shows the improvements in thrust-to-weight ratio that have been made in recent years, and also a projection of what is required in the future. It can be seen that the rate of improvement over the next 15 years must be about twice the current rate. It has been identified that approximately half of the improvements can be achieved with advanced materials, and the other half needs to be achieved by improved component efficiencies and by reducing the air "leakage" from the propulsion stream. Once again advanced CFD codes will be needed to achieve the necessary component performance goals and to ensure that as much as is possible of the air entering the engine goes to generate useful thrust.

The successful applications of CFD codes ob-

tained to date should not give rise to complacency. To the contrary, if the aggressive military and commercial goals needed to keep this nation competitive with outside threats are to be met, then significant advances in the development and application of CFD codes will need to be made. Two major limitations exist that currently are impeding further progress in the development and use of CFD codes. They are: the inadequacy of physical models that need to be incorporated in order to simulate realistic engine flow fields, and the limitations in computer capability associated with speed, memory capacity and cost of calculation.

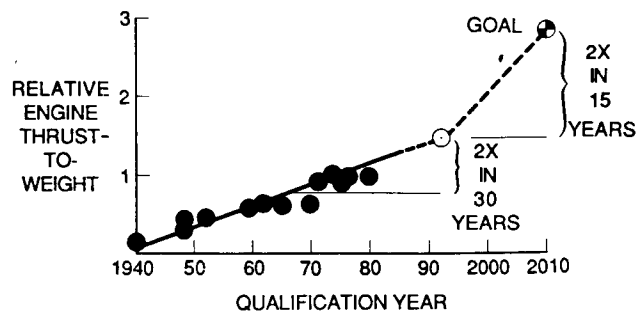


Figure 16 Increase in Engine Thrust to Weight Ratio with Time

The first and perhaps the most obvious deficiency in current CFD codes is the lack of realistic physical models. Empirical data from many different types of test are used to calibrate the codes for application to gas turbine design. For example, in turbomachinery flows, surface shear forces and trailing edge conditions are established in order to obtain the required accuracy in predicting airfoil loadings, air angles and weight flow. Turbulence models that are necessary to provide closure to the Reynolds-averaged Navier-Stokes equations have evolved to the point where relatively simple flows can be simulated, but they are insufficient to be used as a reliable means for predicting airfoil loss and heat transfer. For the reasonably near future, it seems unlikely that the complex flows in airfoil boundary layers and in free shear flows will be calculable. Thus, if CFD codes are to be used in the future to optimize on minimum loss or heat transfer, a large input from key experiments will be required. Similarly, the interactions between turbulence and heat release in the combustor are not well understood and can only be modeled in crude fashion, for which empirical input is required. The problems in generating these data are twofold: what are the key experiments to be performed that are needed to develop codes which can be applied to flows in an engine environment, and how will the large number of these experiments be funded? Attempts at answering these questions are beyond the scope of this paper, but they need to be addressed by all who are in the business of developing and applying CFD codes.

No discussion of the limitations of physical modeling can omit the fact that almost all of the CFD codes used in gas turbine design are steady, and that the so-called Navier-Stokes codes usually do not solve the Navier-Stokes equations, but employ a statistical turbulence management strategy and solve the time-averaged Reynolds or Favre equations. Yet, of course, the flow in most gas turbine components is highly unsteady. In fact, it



is quite remarkable that so much progress in calculating component flows has been made without considering unsteadiness as a first order factor. Again, using an example from turbomachinery flows, the flow in a specific turbine or compressor airfoil passage is very much dependent on the flow field exiting the upstream airfoil row. As discussed in the section on Euler codes, because the upstream row is moving relative to the specific airfoil passage in question, the flow field from the upstream row currently is pitchwise averaged and the flow is assumed steady and circumferentially constant as it enters the airfoil passage that is being analyzed. From the work of Sharma, et al (1985) it can be shown that this simplification results in the suppression of vorticity entering the airfoil passage. Since the function of the airfoil passage is to turn the flow (and the turning angle can be over 100 degrees for turbines), the redistribution of the entering vorticity is instrumental in determining the exit flow angles. Thus, if much of the inflow-vorticity is suppressed, it is not possible to obtain reliable exit flow predictions from an essentially steady CFD code. Similarly, in the combustor, the current calculation of the distribution of fuel across the reaction zone depends on the trajectories of the fuel droplets (which should include the effects of turbulent dispersion of the droplets and modulation of the gas-phase turbulence by the presence of the droplets) and the assumption of gradient diffusion. However, observations of the flow patterns in combustor primary zones show that nonstationary effects can be dominant. Thus, the actual mixing of fuel and air is largely by processes that are not accounted for in the calculation procedure which solves time-averaged equations. One solution to these problems is to develop unsteady three-dimensional CFD codes, and starts have been made in this direction. Such codes introduce such a level of complexity and calculation cost, however, that it is unlikely that unsteady codes will be used significantly in the design of gas turbine components for several years (Sturgess, 1984).

The other significant limitation in developing and applying CFD codes is the current state of the art of computer hardware. In gas turbines, the geometries and the flow fields are highly complex; thus, large numbers of grid points are desired by the code developers in order to predict the flow field with the accuracy required by the designer. The designer also wants to obtain answers fast, make some changes and re-run the code until appropriate design criteria are met. Since designs are generated within a given budget, the designer also wants the cost of each run to be reasonable in order to stay within that budget. The code developer of course also wants inexpensive and fast computing capability so that the parametric studies necessary for calibrating the code for a wide range of geometries and flow conditions can be conducted. Many papers in the literature have realistically assessed the computer hardware needs for the future, e.g., Sturgess (1985a) for the combustor, and the Numerical Aerodynamic Simulator (NAS) has a well-established program in place to meet many of these needs. Within this context, emphasis must be placed on re-configuring CFD codes to be optimally efficient on vector and parallel-processing computer architectures. Emphasis must also be placed on exploiting such techniques as adaptive grid procedures to reduce as much as possible, the computer resources needed for solution of a spe-

cific problem.

## CONCLUDING REMARKS

It has been shown that internal flows for aerospace propulsion are now being extensively calculated in the industry as part of the design and development procedure of engine components. Although the illustrative examples presented have been taken exclusively from the gas turbine engine, this is so for ramjets, scramjets and liquid-fueled rocket motors also. A variety of codes is used, and the physics embodied in these codes range from potential flow to the Navier-Stokes equations. Selection of the appropriate code depends on the component flow required to be calculated and on the objectives required of the calculation. The codes are currently making a useful and important contribution to achieving the required performances of the various engines.

While the utility of the current generation of codes cannot be denied, there are some limitations to the accuracy of the calculations produced, and these were briefly touched upon. These limitations can be succinctly described as: (1) lack of physical realism in the modeling, and (2) restraints imposed by available computer resources.

The Numerical Aerodynamic Simulator (NAS) represents a great resource with immense potential for assisting with the calculation of internal flows in aerospace propulsion engines of all types. It is an opportunity for the present barriers to accuracy in such calculations to be overcome. The NAS, because of its high speed and large storage capacity, can be used as an experimental wind tunnel, or in the present circumstances, as an experimental engine component. In this role, it can be used to make calculations that are free from computer-resource limitations; therefore, it can be used to improve the understanding and modeling of relevant physical phenomena and to provide calibration information that is necessary for simpler, design-oriented codes intended for regular engineering use.

For example, the tip clearance flow field is a major contributor to loss in compressors; but because of the prohibitive number of grid points which would be necessary to incorporate it accurately in a numerical treatment of a full blade row, it is currently represented by very crude empirical modeling. The NAS would permit calculation of the actual tip clearance flow field with sufficient resolution when embedded in a full three-dimensional representation of blade row geometry to lead to more fundamental models.

In the case of non-stationary flows, the capabilities of the NAS can permit the use of unsteady codes for relevant cyclical periods. For turbomachinery applications, blade and vane row interactions can be examined and methods developed to model average flows at computational interfaces. This information can be incorporated in steady flow design codes, which will then more accurately model the interaction effects. Similarly, non-stationary mixing and chemical reaction processes in the combustor could be accurately modeled on the NAS facility, permitting full evaluation of the relative magnitude of the effect of time-dependent processes on the time-average solution.

The examples given above suggest how the NAS might contribute to engineering design capability. How useful the facility actually is will depend on the aerospace propulsion industry learning how to exploit its capabilities to the best advantage. Certainly, it seems apparent that the computational capacity offered by the NAS will be as necessary to achieving the goals of the new engine for the Advanced Tactical Fighter (Petty, et al, 1986) and the lightweight stoichiometric engine, as it is to the Space Shuttle Main Engine and the aerospace plane.

#### REFERENCES

- Clark, D. R.; Maskew, B.; and Dvorak, F. A.: The Application of a Second Generation Low-Order Panel Method - Program VSAERO - to Powerplant Installation Studies. AIAA-84-0122, 1984.
- Huber, F. W.; Rowey, R. J.; and Ni, R-H.: Application of 3D Flow Computation to Gas Turbine Aerodynamic Design. AIAA-85-1216, 1985.
- Kunz, R. F.; and Rhie, C. M.: private communication, 1986.
- Lord, W. K.; and Zysman, S. H.: VSAERO Analysis of a Wing/Pylon/Nacelle Configuration. AIAA-86-1523, 1986.
- Ni, R-H.: A Multiple-Grid Scheme for Solving the Euler Equations, AIAA Journal, vol. 20, no. 11, Nov. 1982, pp. 1565-1571.
- Petty, J. S.; Hill, R. J.; Piccirillo, A. C.; and Fanning, A. E.: The Next Hot Fighter Engine, Aerospace America, vol. 24, no. 6, June 1986, pp. 40-43.
- Sharma, O. P.; Butler, T. L.; Joslyn, H. D.; and Dring, R. P.: Three-Dimensional Unsteady Flow in an Axial Flow Turbine, Journal of Propulsion and Power, vol. 1, no. 1, Jan.-Feb. 1985, pp. 29-38.
- Sturgess, G. J.: Gas Turbine Combustor Liner Durability - The Hot Streak Problem. Gas Turbine Combustor Design Problems, Ed. A. H. Lefebvre, Hemisphere Publishing Company, 1980, pp. 133-150.
- Sturgess, G. J.: Aerothermal Modeling - Phase I, Final Report. NASA CR-168202, May 1983.
- Sturgess, G. J.: Stationary State Computational Fluid Dynamics for Aeropropulsion Devices, Computational Methods, Ed. K. L. Strange, CPIA Publication 401, Feb. 1984, pp. 25-51.
- Sturgess, G. J.; James, R. H. and Syed, S. A.: Computing Gas Turbine Combustors, Proc. Symp. Computers in Engineering 1985, vol. 3, ASME Computer Engineering Division, August 1985a, pp. 261-275.
- Sturgess, G. J.; Syed, S. A.; and McManus, K. R.: Calculation of a Hollow-Cone Liquid Spray in a Uniform Airstream, Journal of Propulsion and Power, vol. 1, no. 5, Sep.-Oct. 1985b, pp. 360-369.
- Sturgess, G. J.; and Datta, P.: Calculation of Flow Development in Rotating Passages (to be published, 1987).
- Ting, C. T.; Kaldschmidt, G.; and Syltebo, B. E.: Design and Testing of New Center Inlet and S-Duct for B-727 Airplane with Refanned JT8D Engines. AIAA-75-59, 1975.